Technology Opportunity

Computer Simulation of Flow Through Ducts

The use of Computational Fluid Dynamics (CFD) for the design and analysis of internal and external flow systems has increased greatly in the last few years. Problems that previously could only be investigated by expensive and time-consuming experimental testing can now be solved on the computer. CFD is already widely used by the large aerospace, automotive, and heating and air-conditioning companies. Until recently, CFD codes were typically run on large main-frame super-computers, but solution algorithms and computational hardware have improved to the point that it is now practical to run these codes on desktop workstations and even PC's. The National Aeronautics and Space Administration (NASA) is seeking additional industrial partners to cooperatively develop, enhance, and validate the CFD codes NPARC and RNS3D.

Potential Commercial Uses

- Analyze existing configurations to trouble-shoot problems
- Investigate proposed changes to existing designs
- Design new systems
- Perform CFD computations for other organizations
- Provide support for other organizations using NPARC and/or RNS3D

Benefits

Commercial CFD codes exist, and may be a good choice for a small business with limited expertise in fluid dynamics. The principal advantages to government-developed codes like NAPARC and RNS3D are

- Programs, documentation, and a limited amount of support are free
- Codes may have advanced features not available commercially
- Source code is supplied

The Technology

The flow of a fluid (gas or liquid) is governed by a complex set of equations called the Navier-Stokes equations. The NPARC code solves the complete set of equations, and hence is applicable to almost any nonrotating, noncombusting, fluid flow problem. NPARC's grid blocking and generalized boundary condition capabilities allow simulation of complex geometric configurations. The flow may be steady or unsteady; two-dimensional, axisymmetric, or three-dimensional; subsonic or supersonic; inviscid or viscous; and laminar or turbulent. Separate pre- and post-processors are needed to create the computational grid and to graphically display the results. NPARC is written primarily in FORTRAN 77, with some C coding, and is normally run on a UNIX mainframe or workstation, although the two-dimensional code may be run on a PC. The static pressure distributions shown in Figure 1 were computed as part of a design study for a new compensated pitot-static tube. NPARC is currently being developed and supported by the NPARC Alliance, a joint effort of the NASA Lewis Research Center and the Air Force Arnold Engineering Development Center.

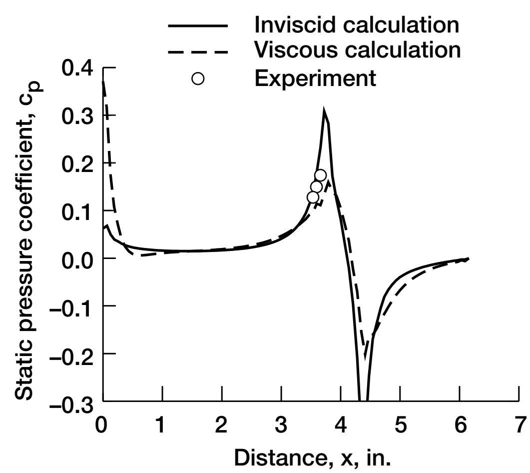
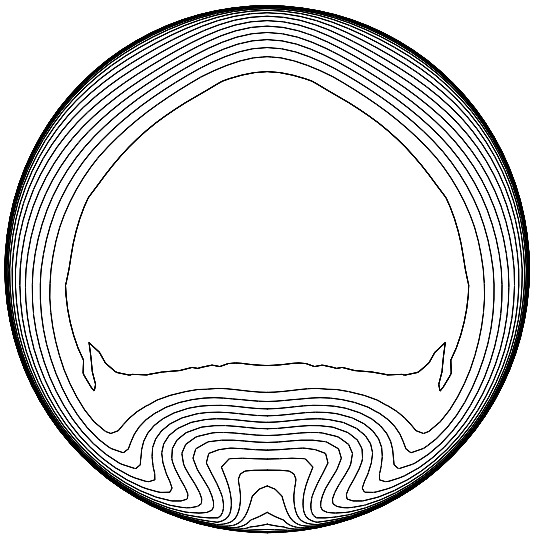
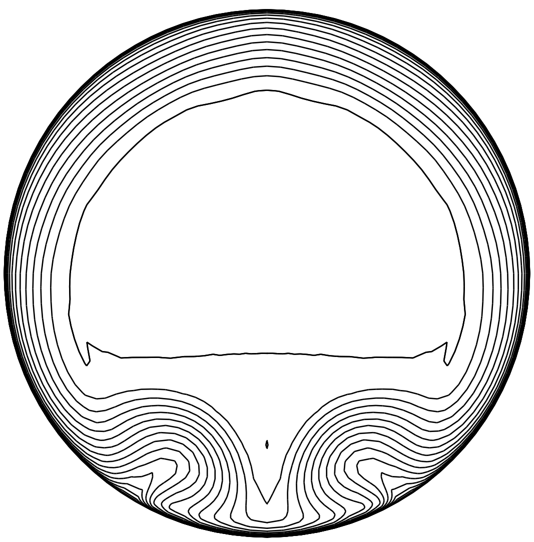


Figure 1.—Surface static pressure distribution for compensated pitot-static tube.

National Aeronautics and
Space Administration
Lewis Research Center



(a) Without vortex generators.



(b) With vortex generators.

Figure 2.—Compressor-face total pressure contours.

The RNS3D code solves a reduced set of the equations and is applicable to subsonic internal duct flows without abrupt changes in geometry. The flow must be steady, viscous, and three-dimensional, and may be laminar or turbulent. The computational grid may be created by a separate pre-processor, or internally by RNS3D; a separate post-processor is needed to graphically display the results. RNS3D is written entirely in FORTRAN 77, and is normally run on a UNIX mainframe or workstation, although it is fast enough to be run on a PC. Figure 2 shows total pressure contours at the compressor face of an S-shaped subsonic inlet for a new business jet. These results were computed during the design of a vortex generator system to lower the compressor-face distortion level. RNS3D is currently being enhanced and supported by the NASA Lewis Research Center.

Options for Commercialization

The National Aeronautics and Space Administration is seeking additional industrial partners to cooperatively develop, enhance, and validate the NPARC and RNS3D codes. NPARC and RNS3D may have redistribution and export restrictions.

Contact

Charles E. Towne
MS 5-11
NASA Lewis Research Center
21000 Brookpark Rd.
Cleveland, OH 44135

Email: towne@lerc.nasa.gov
Phone: (216) 433-5851

Key Words

CFD
Flow simulation
Duct flow
Internal flow
Viscous flow

National Aeronautics and
Space Administration
Lewis Research Center